Inno GATIN, Fakultet strojastva i brodogradnje, Sveučilište u Zagrebu, Ivana Lučića 5, HR-10000 Zagreb, Croatia, ig185258@stud.fsb.hr
Vuko VUKČEVIĆ, Fakultet strojastva i brodogradnje, Sveučilište u Zagrebu, Ivana Lučića 5, HR-10000 Zagreb, Croatia, vuko.vukcevic@fsb.hr
Hrvoje JASAK, Fakultet strojarstva i brodogradnje, Sveučilište u Zagrebu, Ivana Lučića 5, HR-10000 Zagreb, Croatia, hrvoje.jasak@fsb.hr

SIMULATION OF WAVE IMPACT LOADS IN OPENFOAM

Abstract

This paper presents numerical simulation of wave loads on static offshore structures. The Naval Hydro Pack in OpenFOAM is used. Two phase free surface flow model is used with Volume of Fluid and Level Set method for interface capturing. Governing equations are discretized using the Finite Volume Method. Wave modelling with relaxation zones is described. Simulation of 3D dam break with fixed square column is carried out. Results are compared with corresponding experimental data. Next, a method for freak wave initialization using wave focusing is given. Wave components are obtained using standard wave energy spectra. Simulation of freak wave is shown.

Key words: OpenFOAM, finite volume method, wave impact loads, relaxation zones, freak wave

SIMULACIJA UDARNIH VALNIH OPTEREĆENJA U OPENFOAM-U

Sažetak

Tema ovog rada je primjena numeričkih proračuna udarnih valnih opterećenja na nepomične konstrukcije pomorske tehnike. Korišten je programski paket Naval Hydro Pack u programu OpenFOAM. Korišten je model dvofaznog strujanja sa slobodnom površinom pomoću *"Volume of Fluid"* metode i *"Level Set"* metode. Jednadžbe su diskretizirane metodom kontrolnih volumena. Ukratko je prikazan način modeliranja valova pomoću relaksacijskih zona. Proveden je proračun opterećenja kvadratnog stupa uslijed udarnog vodenog vala te su rezultati uspoređeni s eksperimentalnim vrijednostima. Prikazan je način generiranja ekstremnog vala (engl. *„freak wave"*) pomoću fokusiranja valnih komponenti standardnog spektra energije valova. Provedena je simulacija ekstremnog vala te su prikazani rezultati proračuna.

Ključne riječi: OpenFOAM, metoda kontrolnih volumena, udarna valna opterećenja, relaksacijske zone, ekstremni val

1. Introduction

Calculating wave loads on offshore structures and ships is not an easy task. For many cases it is not possible to accurately predict loads that might occur during service. This is why Computational Fluid Dynamics (CFD) methods are getting more attention in order to acquire more precise information about wave loads. The phenomenon of freak wave is especially dangerous for offshore structures, and conventional methods are not adequate for calculation of freak wave loads.

This paper is organized as follows. In section 2 governing equations for two phase flow are given and method for wave generation is presented. This includes Navier-Stokes equations and interface tracking methods, Volume of Fluid and Level Set. Next, in section 3 some details of numerical procedure are given. Section 4 presents the results of validation cases. Two simulations are conducted in this paper. Finally, conclusion is given discussing the results of simulations.

2. Governing equations

In this section mathematical model for incompressible two-phase flow is presented. Two methods used for interface capturing are Volume of Fluid (VOF) [1] method and Level Set (LS) [2] method. Also, wave modelling using relaxation zones is described.

2.1. Continuity and Navier – Stokes equations

Governing equations for incompressible, two-phase flow are continuity equation (1) and Navier-Stokes equations (2):

$$\nabla \cdot \mathbf{U} = 0, \tag{5}$$

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U}\mathbf{U}) - \nabla \cdot \left(\mu_{eff} \nabla \mathbf{U}\right) = -\nabla p + \rho \mathbf{g} + \nabla \mathbf{U} \cdot \nabla \mu_{eff} + \sigma \kappa \nabla \alpha, \tag{6}$$

where $\mathbf{U}$ is the velocity vector, p pressure, ρ and μ_{eff} are density and dynamic viscosity, respectively. $\mathbf{g}$ is the gravity force, σ and κ present the surface tension coefficient and mean free surface curvature, respectively [1]. α is the volume fraction variable which will be described in the following section.

2.2. Volume of Fluid method

This method is based on indicator function α, which is used to determine fluids viscosity and density using following expressions:

$$\rho = \alpha \rho_1 + (1-\alpha)\rho_2, \tag{7}$$

$$\mu = \alpha \mu_1 + (1-\alpha)\mu_2. \tag{8}$$

α function is bounded between 0 and 1. It has the value of 1 in water and value of 0 in air. Transport equation for α function is:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) + \nabla \cdot (\alpha(1-\alpha)\mathbf{U}^{\mathrm{r}}) = 0. \tag{5}$$

The third term is responsible for interface compression [3] which is used to reduce the smearing α function due to numerical diffusion, thus preserving sharp interface.

2.3. Level Set method

This method is based on level set function ϕ, that is signed distance function [4]. Main advantage of this approach is the continuity of the ϕ function, which enables easier numerical implementation and sharper interface. Absolute value of signed distance function is equal to the nearest distance from the observed fluid particle to the free surface. Values of the function are positive in one fluid, while in the other values are negative. Free surface is defined as the zero isocontour of the signed distance function ϕ. Transport equation for the level set variable is:

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \mathbf{U}) = 0. \tag{6}$$

After the transport, redistancing equation is introduced to preserve signed distance function:

$$\frac{\partial \phi}{\partial \tau} + S(\phi_0)(|\nabla \phi| - 1) = 0, \tag{7}$$

where τ presents pseudo time that is used for iterative solution process. More details on the subject can be found in [4].

Implementation of the Level Set method in OpenFOAM is described in [2]. Modifications are made to improve the method, i.e. to make it conservative. For this purpose, iterative addition of volume is implemented to accommodate the volume loss.

2.4. Wave modelling using relaxation zones

Wave modelling is conducted with coupling of potential flow wave theories and CFD solutions. Relaxation zones are used [5] to smooth out the transition between wave theory and CFD calculations. This method also uses outlet relaxation zone, which includes constant velocity on the whole depth of the relaxation zone. The variables are coupled implicitly using weight functions f_R as follows:

$$\phi = f_R\, \phi_{potential} + (1 - f_R)\phi_{CFD} \tag{8}$$

3. Numerical procedure

PIMPLE algorithm is used for pressure-velocity coupling. PIMPLE is an algorithm that uses a few PISO correctors per each SIMPLE corrector. As well as momentum and pressure equations, VOF or LS transport equations are also treated implicitly. Implicit solving enables higher Courant numbers. In this paper two SIMPLE correctors and four PISO correctors are used per each time step. Second order accuracy in space is achieved. For temporal discretization Euler implicit scheme is used, and for convective terms van Leer scheme is mostly used [6].

4. Test cases

In this section two validation cases are presented. The first test case is a simulation of 3D dam break impacting on a square cross section column. Forces are calculated and compared with experimental results. The second test case consists of a simulation of a freak wave.

4.1. 3D dam break simulation

Dam break on a square column is a very transient phenomenon with violent free surface effects. Reason for this are the sharp edges of the column. Simulation is conducted based on the experiment described in [7]. Geometry of the simulation domain completely corresponds to experimental set-up shown on Fig. 1. All dimensions in Fig. 1. are in meters. Simulation domain with position of free surface at time zero is shown in Fig. 2.

Two simulations are carried out using different mesh resolutions. Level Set method is used to capture the position of the interface because it proved to handle violent free surface effects somewhat better than VOF method. With VOF method, force was poorly predicted due to unacceptable smearing of the interface. Tab. 1 shows relevant simulation parameters for the two simulations.

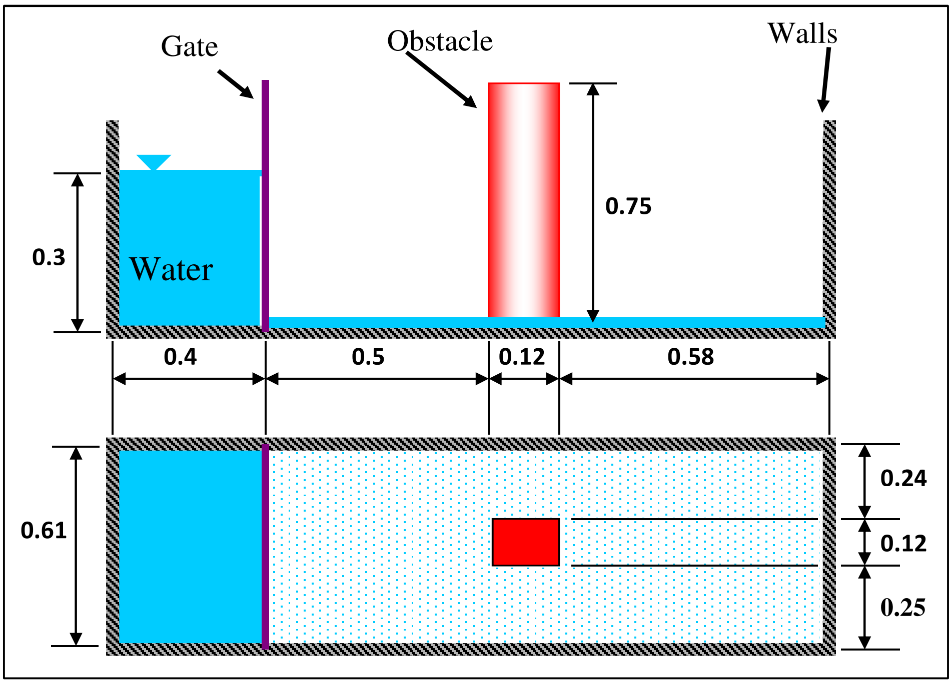

Fig. 1. Experimental tank geometry [7].

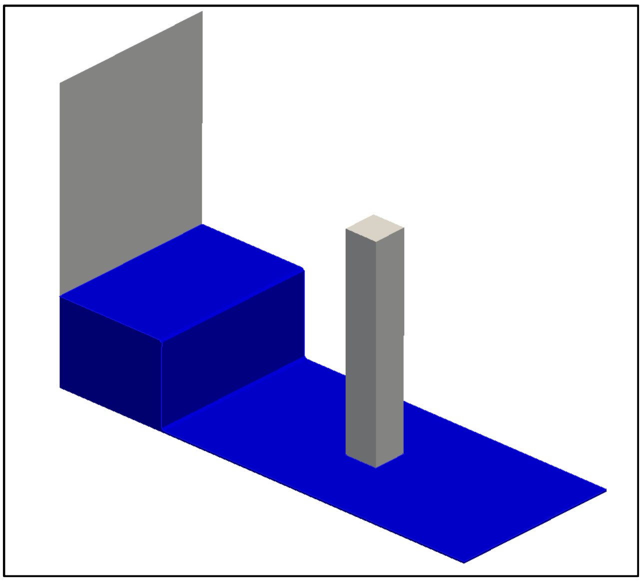

Fig. 2. Simulation domain layout.

Table 4. Simulation parameters

Simulation number	1	2
Number of cells	1 000 000	4 600 000
Courant number	0.5	2
α Courant number	0.25	1
CPU time, h	10	27
Processor speed, GHz	4 x 3.4	4 x 3.7
RAM, GB	15	16
Δt, s	0.00093	0.00090

Load forces on the column in the longitudinal direction are calculated and compared to the experimental forces. Results show very good agreement with available experimental data, shown on Fig. 3. Data depicted with red crosses are experimental results, and continuous lines present simulation results. It can be seen that both simulations describe the force signal accurately. The first force peak occurred somewhat earlier when compared to experiment. This is due to the fact that the gate, which is used in the experiment to release the mass of water, is not simulated.

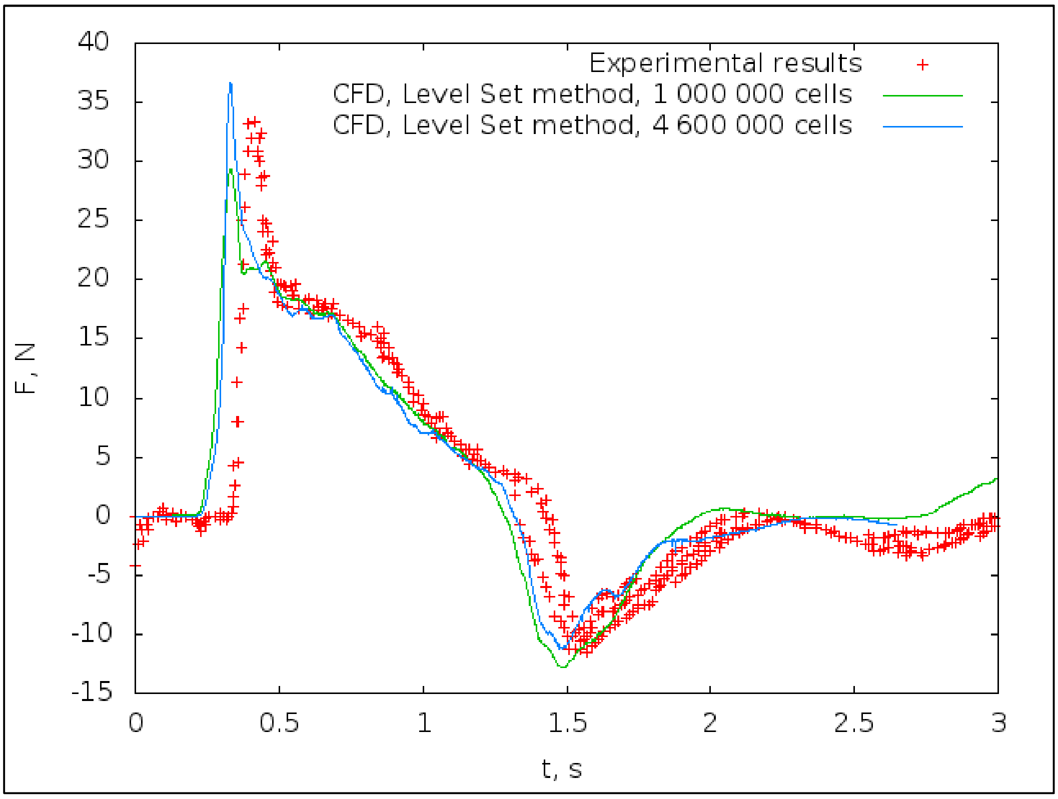

Fig. 3. Forces exerted on the column in the direction of water wave propagation.

4.2 Freak wave simulation

Freak wave phenomenon is becoming more important when it comes to offshore structures. According to the most widely accepted definition, a freak wave, also known as a rogue wave, is a wave whose height exceeds the significant wave height of the given sea state by at least a factor of two [8].

In this study, wave focusing method is used to generate a freak wave. Thirty harmonic wave components are used for focusing guided by the recommendation in [9]. Pierson-Moskowitz sea energy spectrum is used to determine the harmonic components in order to obtain a more realistic wave. Phase shifts of the individual components are determined by optimization in order to achieve positive superposition of wave components in the desired location and time frame. It was not possible to obtain phase shifts as uniformly random numbers since it would take a long time for the freak wave to occur in the desired position. The reason for this is low probability of occurrence of the freak wave in realistic sea state which is approximately 10^{-8} [8]. The obtained wave profile has the height of 0.255 m, which is 2.12 times higher than the selected significant height of the sea state of 0.12 m. This qualifies this wave as a freak wave. Simulation is carried out in a wave flume which has a vertical truncated cylinder immersed in water. Fig. 4 depicts the freak wave profile in the simulation shortly before the focus time of 2.66 s. Blue colour shows the part of the domain which contains water ($\alpha = 1$), red colour shows the air in the domain ($\alpha = 0$). Green color shows the transitional area between air and water, i.e. values of α are between 0 and 1. Thick horizontal white line in Fig. 4 is positioned at calm free surface level. Thinner white horizontal lines show positive and negative significant amplitudes, i.e. distance between them is the significant wave height. Black horizontal lines are positioned at the bottom and top of the freak wave. Height difference is obvious, and it agrees well with calculated height. The white vertical line shows the position of the cylinder centreline. Fig. 5 shows a frame shot of simulation at the same moment of time as in Fig. 4. Further propagation of freak waves caused further steepening of the front wave slope. Fig. 6 shows frame shot in the moment of impact against the vertical cylinder. It can be observed that that the wave slope was close to 90 degrees at moment of impact. That is in accordance with often encountered description of freak wave as a “wall of water ”.

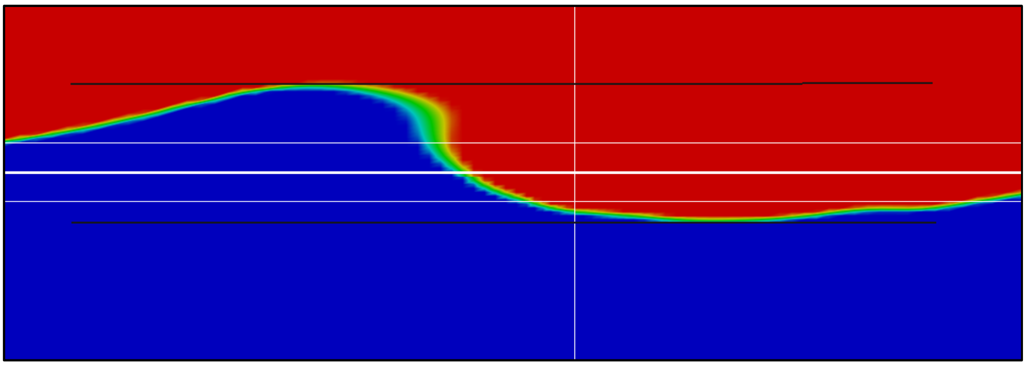

Fig. 4. Freak wave profile during simulation.

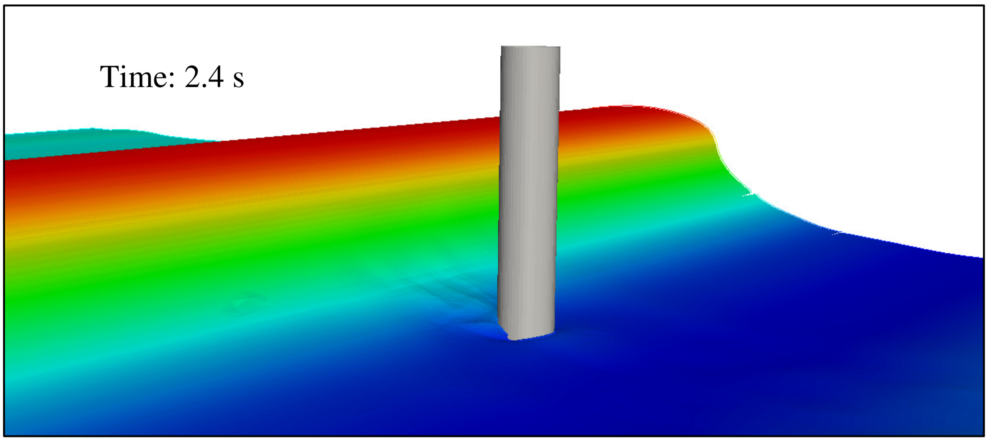

Fig. 5. Simulation frame shot in the moment before the wave impact.

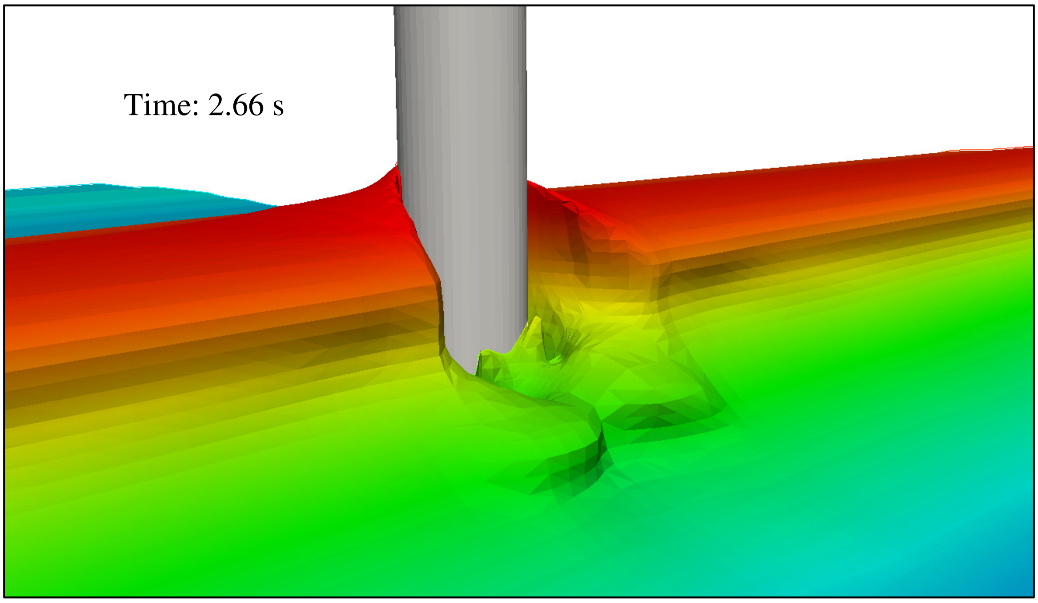

Fig. 6. Simulation frame shot in the moment of wave impact.

Force in the direction of wave propagation exerted on the cylinder is shown on Fig.7 in respect to time. Force peak can be seen in the moment of freak wave impact. Experimental data is not available for comparison. It is obvious from Fig. 6 and 7 that this is a highly nonlinear phenomenon which can hardly be described using conventional potential methods.

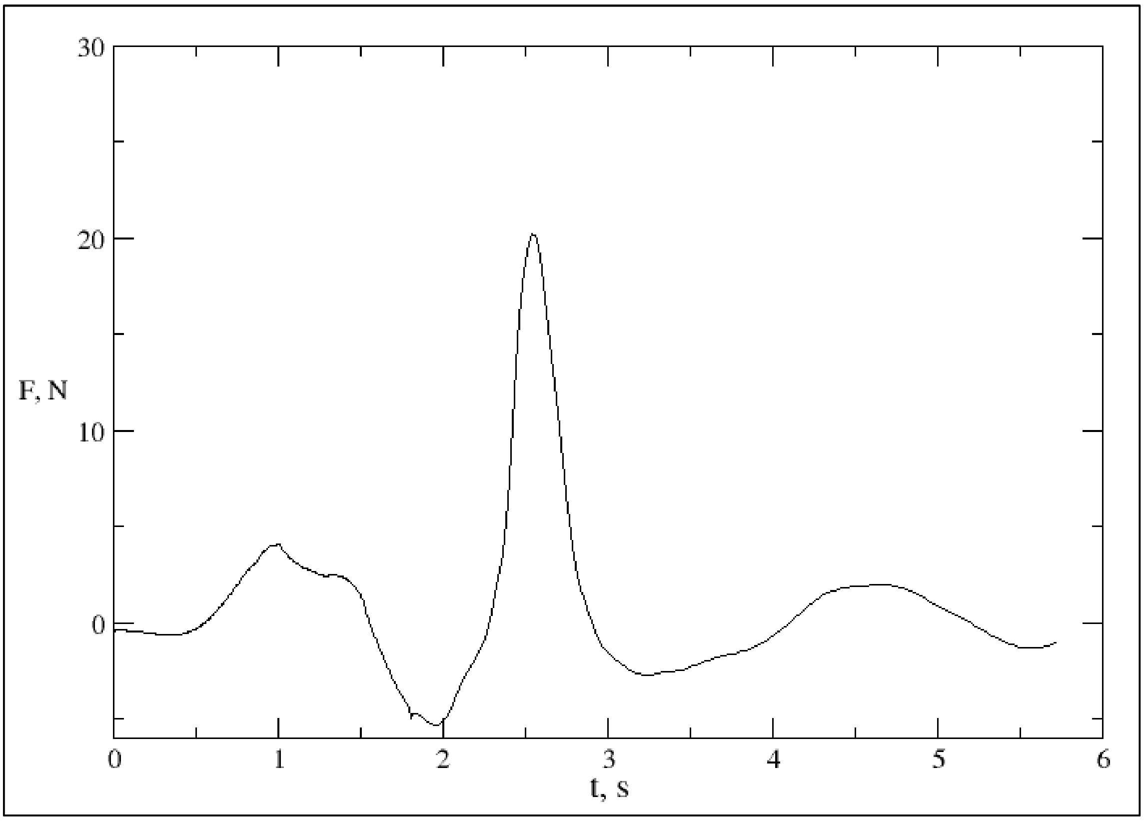

Fig. 7. Force exerted on the cylinder.

5. Conclusion

In this paper two test cases are presented, showing the applicability of CFD simulations in the field of offshore engineering and naval architecture. Finite Volume Method is used for numerical simulation. Volume of Fluid and Level Set methods are used for interface capturing.

The first test case shows that excellent results can be obtained on coarse mesh in several hours of simulation time for cases of wave water impact. This is applicable for calculating green sea loads on exposed deck structures and similar problems. Advantages over experimental methods are obvious, while other methods are not easily applicable for this kind of problems. The second test case depicts the possibility of freak wave modelling in CFD which can be used for calculating extreme loads on offshore structures.

References

[1] UBBINK, O.: “Numerical prediction of two fluid systems with sharp interfaces”, Imperial College of Science, Technology & Medicine, London, 1997.

[2] VUKČEVIĆ, V., JASAK H.: “Finite volume simulation of two phase flows using a conservative level set method for interface tracking”, 11.th World Congress of Computational Mechanics, work in progress, 2014.

[3] RUSCHE, H. “Computational Fluid Dynamics of Dispersed Two Phase Flows at High Phase Fractions”, Imperial College of Science, Technology & Medicine, London, 2002.

[4] OSHER, S., FEDKIW, R.: “Level Set Methods and Dynamic Implicit Surfaces”, Springer, New York, 2003.

[5] JACOBSEN, N. G., FUHRMAN, D. R., FREDSØE, J.: “A Wave Generation Toolbox for the Open–Source CFD library: OpenFoam”, International Journal for Numerical Methods in Fluids 9(2012), p.1073 – 1088.

[6] VAN LEER, B.: “Towards the Ultimate Conservative Difference Scheme. IV. A New Aproach to Numerical Convection”, Journal of Computational Physics 23(1977), p. 276-299.

[7] GASTEIRA, M.: “Test 1- Force exerted by a schematic 3D dam break on a square cylinder”, [Online] Available at http://cfd.mace.manchester.ac.uk/sph/TestCases/SPH_Test1.html, 20.3.2014.

[8] KHARIF, C., PELINOVSKY, E.: “Physical mechanisms of the rogue wave phenomenon”, European Journal of Mechanics - B/Fluids 22(2003), p.603-634.

[9] ZHAO, X.: “Numerical simulation of extreme wave generation using VOF method”, Jurnal of hydrodynamics 22(2010), p.466-477.